NGSPICE Tutorial

By
Yogesh Mahajan and
Shashank OV

Bridge Courses, IIT Bombay (July, 2018)

What is NGSPICE ?

- Spice is an acronym for the **S**imulation **P**rogram with **I**ntegrated **C**ircuit **E**mphasis. First developed by UC Berkeley. It is the origin of most modern circuit simulators.

- NGSPICE is an open source mixed signal/mixed level simulator. It is the result of combining SPICE with some extra analysis, modeling methods and device simulation features.

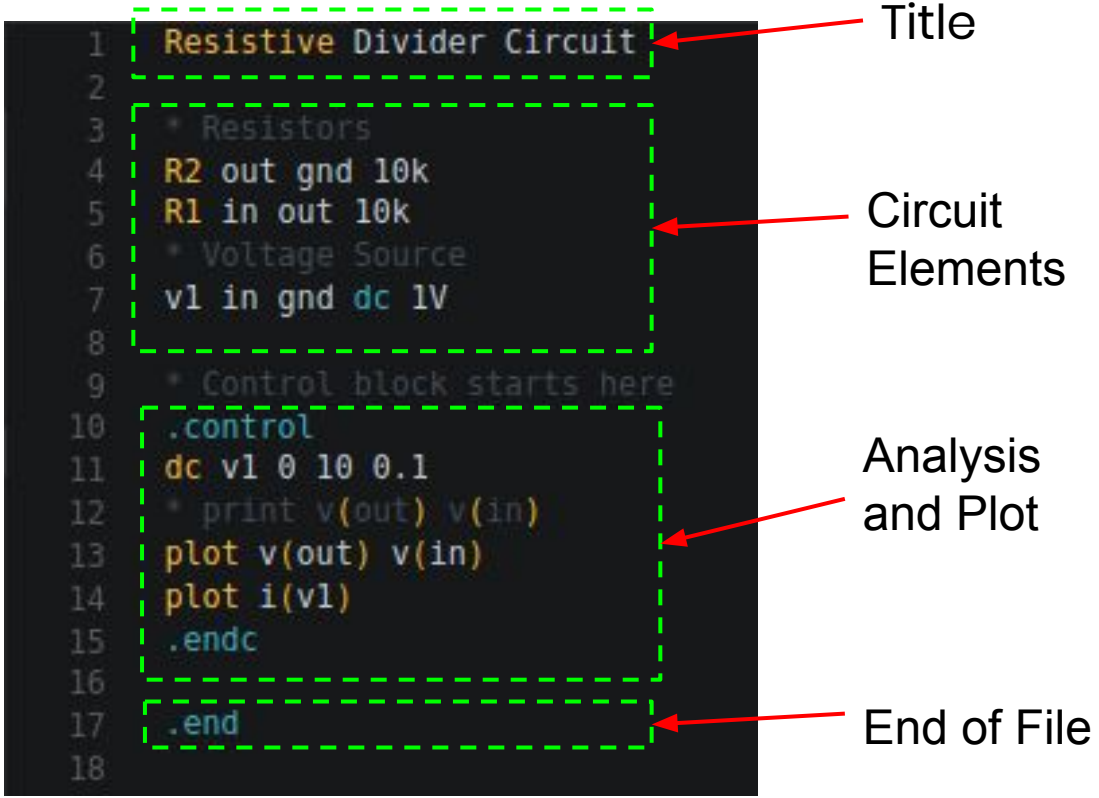
- It is available for Linux, MAC and Windows. *It is recommended to use Linux system for NGSPICE*.

- NGSPICE requires a circuit described in netlist format. A *netlist* is defined as a set of circuit components and their interconnection. It may also include simulation controls.

NGSPICE provides you with …

- Basic circuit elements
  - Passive components: resistors, capacitors, inductors, etc.
  - Sources: voltage and current sources, controlled sources
- Semiconductor devices
  - Pre-defined circuit elements such as diodes, BJTs, MOSFETs, etc.
  - Allows user to add device specific models for more complicated devices such as OpAmps and specialized transistors (technology dependent models)
- Circuit analysis techniques
  - DC, AC (small signal) analysis
  - Transient analysis
  - Noise, Pole-Zero analysis and more
- Plot, save and measure analysis results

Getting started with NGSPICE



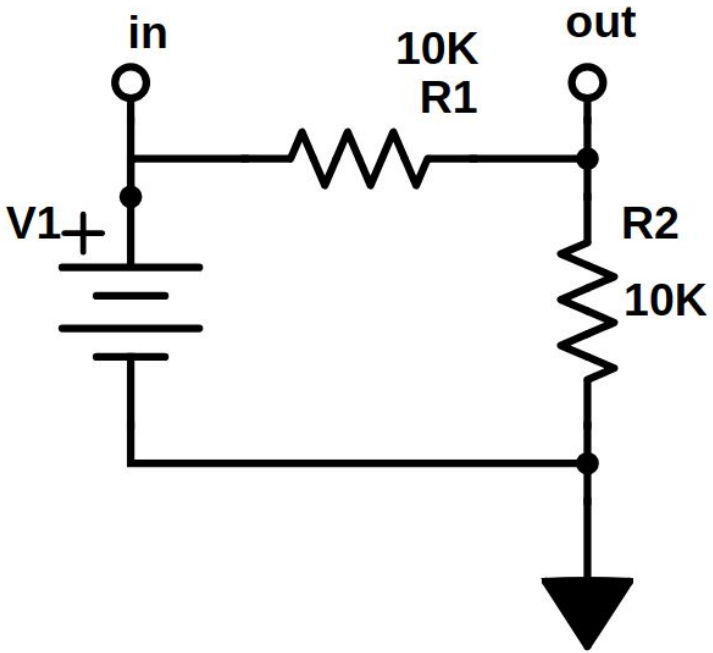
Common Circuit Elements

First letter	Element description
C	Capacitor
D	Diode
F	Current-controlled current source (CCCs)
G	Voltage-controlled current source (VCCS)
I	Current source
J	Junction field effect transistor (JFET)
L	Inductor
M	Metal oxide field effect transistor (MOSFET)
Q	Bipolar junction transistor (BJT)
R	Resistor
X	Subcircuit (for details see below)
Z	Metal semiconductor field effect transistor (MESFET)

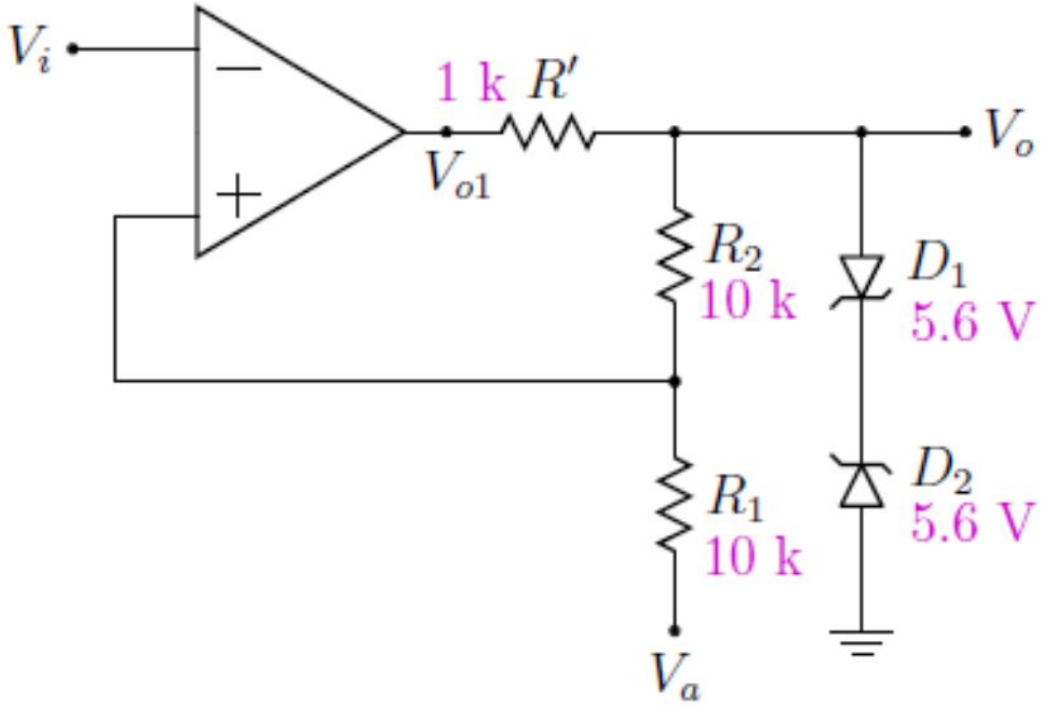
Scaling Factors

Suffix	Name	Factor
T	Tera	10^{12}
G	Giga	10^{9}
Meg	Mega	10^{6}
K	Kilo	10^{3}
mil	Mil	25.4×10^{-6}
m	milli	10^{-3}
u	micro	10^{-6}
n	nano	10^{-9}
p	pico	10^{-12}
f	femto	10^{-15}

Let’s start with a simple circuit



Schmitt Trigger



Thank You!

You can contact us at

Yogesh Mahajan
y.mahajan456@gmail.com

Shashank OV
shashankov@iitb.ac.in

Modified Nodal Analysis

$$\begin{bmatrix} A_{rG} G A_{rG}^{\top} & A_{rT} \\ D A_{rT}^{\top} & C \end{bmatrix} \begin{bmatrix} v_n \\ i_T \end{bmatrix} = \begin{bmatrix} -A_{rJ} i_J \\ s_T \end{bmatrix} \tag{2.25}$$

For an electrical network having conductances, current sources and voltage sources, MNA equations can be written as

$$\begin{bmatrix} A_{rG} \mathbf{G} A_{rG}^{T} & A_{rE} \\ A_{rE}^{T} & \mathbf{0} \end{bmatrix} \begin{bmatrix} v_n \\ i_E \end{bmatrix} = \begin{bmatrix} -A_{rJ} i_J \\ v_E \end{bmatrix} \tag{2.26}$$

Reference:
Application of the fast nonlinear DC Analyzer to Min Cost Flow and Single Source Shortest Path Problems - Gaurav trivedi, H. Narayana, S. B. Patkar
Chapter 2